STUDY OF CNC PROGRAMMING & MACHINING

and

MANUFACTURING OF AVIONICS MOUNTING BRACKET

PROJECT REPORT

Submitted by

PRANAV REDDY G

2017UME0110

in partial fulfillment for the award of the degree

of

BACHELOR OF TECHNOLOGY

in

MECHANICAL ENGINEERING

DEPARTMENT OF MECHANICAL ENGINEERING

INDIAN INSTITUTE OF TECHNOLOGY JAMMU

JAMMU, J&K - 181 221

2019

ACKNOWLEDGEMENT

The internship opportunity I had with HAL, Hyderabad was a great chance for learning and professional development. I am grateful to **Dr S Malla Reddy**, Director, HAL and **Shri DVS Raju**, AGM (HR), HAL, Hyderabad for providing me this opportunity.

I thank the **Training & Placement office, IIT Jammu** for giving necessary advice and guidance.

I am using this opportunity to express my deepest gratitude and special thanks to my project guide, **Shri Ram Kiran** who in spite of being extraordinarily busy with his duties, took time out to hear, guide and keep me on the correct path and allowing me to carry out my project at their esteemed organization.

I would like to thank all the technical staff of the HAL in different departments on the shop floor, who, though busy, gave immense practical insight into their respective shops.

Last, but not least, I am deeply indebted to my parents who have been the greatest support while I worked day and night for the project to make it a success.

I perceive this opportunity as a big milestone in my career development. I will strive to use gained skills and knowledge in the best possible way, and I will continue to work on their improvement, in order to attain desired career objectives.

sincerely,

Pranav Reddy G

LIST OF ABBREVIATIONS

CNC – Computer Numerical Control
NC - Numerical control
DNC – Direct Numerical Control.
CAD - Computer Aided Design
CAM - Computer Aided Manufacturing
G&M Codes - Geometric and Machine Codes

LIST OF TABLES

LIST OF FIGURES

TABLES OF CONTENT

ABSTRACT

Motivation for the work:

Aircraft consist of several avionic components in which some are very delicate and needed to be handled very carefully. A well casing and precise manufacture of the components is necessary to carry them. Brackets are architectural elements. In mechanical terms, a bracket is any intermediate component for fixing one part to another, usually larger, part. What makes a bracket a bracket is that it is intermediate between the two and fixes the one to the other. In our project we are going to manufacture an avionics mounting bracket. Using CNC provides us the precise cutting for the complex design of the mounting bracket to fit exactly and provide strong casing. It is one of the most cost efficient processes used by the industries now-a-days.

Problem statement:

- To study the drawing and make its CAM model using MASTERCAM.
- Selection of tools that can withstand the rigidity and hardness of the material provided.
- Determine the way of cutting and the order of the tool paths so that the dimensions are accurately maintained.
- Obtaining right angle cuts for the given material.
- Use of coolant and cold working of the job which leads to accumulation of unwanted stresses.
- Availability of the appropriate tool for milling.
- Clamping of the job - since it is a multipart job but has two hollow pockets. vice or vacuum clamping to be used and precautions to be taken while using any of them.
- Quadrant placing of the job in the machine.
- Memory limit of the machine - since the program generated should be edited in such a way that it meets the machine memory requirements.
- avoiding wastage of material.

Objective of the work:

The main objective of the work is to understand the different types of CNC machines available in the market that are used by the industries, various operations that can be performed using them, different types of control systems used in CNC, studying the component drawing and the raw materials used, understanding all the types of tools used, maintenance of the machine, to learn CNC programming using Mastercam software, code generation and posting, using CIMCO for program editing, analyzing the back plot, transferring the program to machine, program editing on machine, material clamping on the machine, type of Jigs and Fixtures to be used, coolant types used, to identify a machine that is well suited for our design and to manufacture the component with precise cutting in the cost efficient way.

CHAPTER 1

INTRODUCTION

CNC or computer numerical control is used to automate machine tools that are operated by precisely programmed commands encoded on a storage medium, unlike controlling manually via hand wheels or levers, or mechanically automated with the cams alone.

Computers play an integral part of the control. In modern CNC systems, CAD and CAM programs are used for development and machining of the final finished product. The series of steps needed to produce any part is highly automated and the result closely matches the original CAD design. modern CNC mills differ little in concept from the original model built at the MIT in 1952.

3 Axes Mills typically consist of a bed that moves in the X and Y axes, and a tool spindle that moves in the Z (depth). The position of the tool is driven by motors through a series of step down gears, or in modern designs, direct - drive stepper motor or servo motors, in order to provide highly accurate and precise movements.

Open loop control works as long as the forces are kept small enough and seeds are not too great. On commercial metalworking machines closed loop controls are standard and required in order to provide the accuracy, speed, and repeatability that is often demanded.

As the controller hardware evolved, the mills also evolved. one change has been to enclose the entire mechanism in a large box as a safety measure, often with additional safety interlocks to ensure the operator is far enough from the working piece for safe operation.

Most new CNC systems built today are completely electronically controlled. CNC like systems are now used for many processes on the shop floor that can be described as a series of movements and operations. these include laser cutting, welding, friction stir welding, ultrasonic welding, flame and plasma cutting, bending, spinning, hole punching, pinning, gluing, fabric cutting, sewing, tape and fiber placement, routing picking and placing (PNP) and sawing,.

Most CNC machines use Siemens or Fanuc made control systems. For Fanuc generally the coding is done using G and M codes. G codes are used for machining operations and movement of the tools whereas M codes are used for controlling the spindle movement like on/off, rotation in clockwise / anti - clockwise direction and also for tool change operation. The technological improvements in the CAD/CAM software affect the productive time of the industries to a large extent. One of the most important, cost efficient and easy way of manufacturing the aircraft components precisely is using CNC machines.

With the skilled crew and machine the mass production of aircraft components became easy and efficient. CNC is computer aided programming and the working of machine basically depends on the program given by the administrator. The program generated by the administrator is editable both in computer and directly on the machine.

CNC machining is a manufacturing process in which pre-programmed computer software dictates the movement of factory tools and machines. This process can be used to control a range of complex machining from grinders and lathes to mills and routers. Three dimensional cutting tasks can be accomplished in a single set prompt.

CNC supersedes the limitations of manual control where live operators are needed to prompt and guide the commands of machinery tools via buttons and wheels, which has a high risk of errors.

When the CNC system is activated the desired cuts are programmed into the software and dictated to corresponding tools and machinery which carry out dimension tasks as specified like a robot.

With a numeric control machine programs are inputted via punch cards. The programs are fed to computers through keyboards and programming is retained in a computer memory. The language behind CNC is often referred as G-code.

There are different types of CNC machines available in the market for different purposes and the usage depends on the type of job being performed.

CHAPTER 2

PROJECT DESIGN AND IMPLEMENTATION

APPROACH

- Studying the drawing sheet of the component provided.
- Replication of the drawing in the system using Mastercam.
- Providing the tool paths for various operations like drills, pockets.
- editing of subprograms, generated by each operation when posted in CIMCO, where we can visualize the block spot of the operation.
- Based on all the sub programs a main program is created which is then transferred to the CNC machine for component manufacturing.
- CNC runs as per the program provided and direct changes/modifications can be made on machine with the help of digital display assist.
- The display unit displays the operational program at that moment.
- Finally, the component is manufactured on the CNC.

USING THE SOFTWARE - MASTERCAM

CNC machines use a special programming language called GN-code. MasterCAM is a software that allows users to create GN-code programs that can be used to cut different geometric shapes on CNC machines. Here we study MasterCAM for CNC Milling machines.

The main functions are:

1) DESCRIBE THE GEOMETRY OF THE PART TO BE MACHINED

   You can create the geometry in one of two ways:

   (a) by using the graphical design interface provided by MasterCAM

   (b) by making the design in a CAD software like CATIA, SolidWorks and saving it in a format that can be imported on MasterCAM.

2) CREATE THE TOOL DATABASE

   The tool database describes the geometry of tools available for use in the workshop. The path that a tool travels in order to cut a shape depends on the size of the tool; some other information about the tool is also important – for example, the length of the cutting teeth on a drill or an end mill constraints the depth of the hole these can cut.

   The tool can be selected from the existing library or a new tool can be created by giving appropriate dimensions.

3) CREATING THE G-N CODE PROGRAM (MACHINING PLAN) TO CUT THE PART

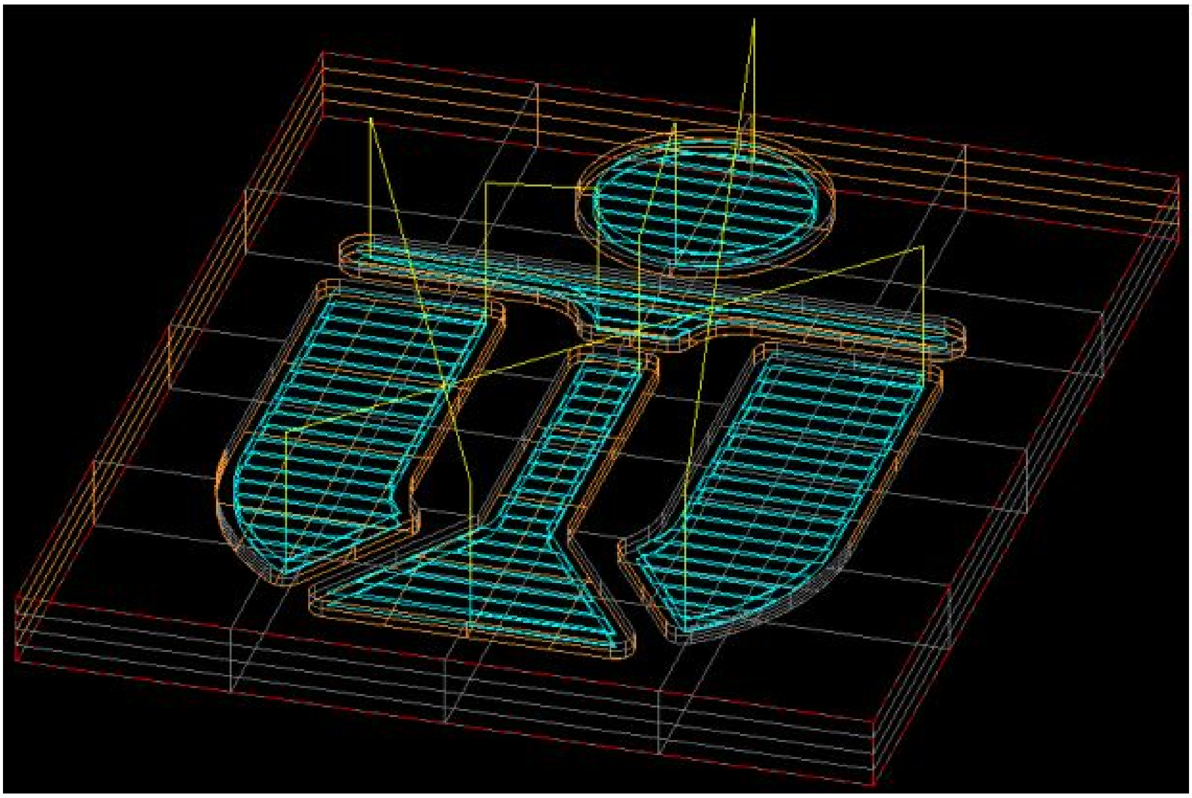
There are many different options and settings in generation of the tool path. Some of them are contour, drill, pocket, flowline, surface. the options are chosen according to the operation to be done on the job.

4) SIMULATE THE MACHINING OF THE PART

It is done for visual verification of the program. You can see a 3D animation of the cutting by the following

MAIN MENU ->Toolpaths -> Operations. The operations Manager window pops up. click on Verify, and then click the Machine button on Verify. the simulation window pops up. You can speed up the simulation by moving the slider to the right. After the simulation is over, the machined part is displayed.

(the simulation of the tool paths look like this Fig. 1)



5) Upload the program to the CNC machine controller.

The tool path generated by MasterCAM has all the information required to control a CNC machine tool to cut the part. However, different CNC machines use slightly different versions of GN-code. The conversion of the machining data to the GN-code specific for a particular CNC machine is called Post-Processing. The exact format of the GN-code is stored in different post-processing files, and the system will use whichever postprocessing format you select.

6) As a last step, the generated code is edited according to the memory requirements of the machine and sent to it.

STRUCTURE OF THE PROGRAM

These are a set of instructions that are given to a CNC machine to control its operations.

- Name of the program.
- Selection of work plane.
- Where Work origin is taken.
- Tool changing position.
- First positioning (movement in working plane) and second positioning (movement in spindle axis) for working.
- Spindle rotation starts and coolant is turned on.
- Third positioning for working (for mechanizing, tool movement with tool radius compensation).
- Depth of cut (in feed only).
- Definition of macro instructions, special commands, tool radius compensation, tool length compensation.
- Cancellation of fixed cycles (canned cycles).
- Return to second position where the spindle stops and coolant is turned off.
- Return to tool change position.
- Movement in spindle axis and working plane.
- End of part program.

PROGRAMMING PROCEDURE

There are two types of data processing techniques:

a) Manual Programming
b) Computer Aided Programming

MANUAL PROGRAMMING: -

In manual programming, the data required for machining a part is a standard format on a special manuscript. The manuscript is a list of instructions, which describe the operations necessary to produce the part. Nearly all the NC systems can be or ginned manually. Part programming of simple parts can be programmed manually with calculators to assist in trigonometric calculators.

Therefore, manual programming is generally for parts to be produced on Point-to-Point machines in which tool path calculations are straightforward and cutter radius compensation are not programming tasks as drilling or straight cutting. In order to apply this procedure, the code by which the functions are represented must be known to the part programmer. To prepare a part program, the planner will decide the order of machining operations needed to program sheet. Each line of program will be numbered in sequence, details of operation are stated and the X, Y and Z coordinates are given. Code numbers representing then supplements the basic information.

1. Preparatory Functions
2. Feed Rate
3. Spindle Speed
4. Required tool
5. Miscellaneous Functions

COMPUTER AIDED PROGRAMMING

With the increased use of NC and growth of complexity of parts to be machined, the part programmer was no longer able to calculate the required tool paths efficiently. It now seems that the use of computers as an aid to part programming is essential. Computers can perform mathematical calculations quickly and accurately. Since the computers have a high degree of reliability, the calculation errors, so common in manual calculations have practically been eliminated. The computer allows the economical programming of tool path of complex parts that could not be programmed manually.

Languages, which are acceptable to general-purpose computer, were developed. These are based on English words and Mathematical notation and are therefore simple to use.

The APT (Automatically Programmed Tools) system is the most comprehensive language for part programming. To program a part in terms of points, straight lines and circles or curves, move definition of cutter is defined by programmer. The computer calculates the tool path, the feed rates, the offsets etc. The result of this computer processing is file of data, which is translated by a post processor program to the coded instructions necessary to operate the NC Systems.

In addition to APT, they are about 20 programming systems that have been developed. More commons are ADAPT, EXAPT, AUTOSPOT, AUTOPROMPT, SPILT and FAPT.

Since HAL procured JAPAN made computer and programming system, the language being used is FAPT. The FAPT (FANUC AUTOMATIC PROGRAMMING TOOL) system is very simple and easy to follow.

GENERAL PROGRAMMING FEATURES OF CNC SYSTEM

There are two ways in which the coordinates data are input to the machine control.

Absolute System: An absolute system is one in which all moving commands are referred to one reference point, which is the origin or is called zero point.

Incremental System: An incremental system is one in which a reference point to the next instruction is the end point of the preceding operation.

DNC SYSTEM: -

When the battery of NC or CNC machine tools is placed under control of a single computer, the result is a system known as Direct Numerical Control. The DNC is also used for management and inventory control, which enhance the use or utilization of computers.

COORDINATE MEASURING MACHINE (CMM)

The Coordinate Measuring Machine (CMM) is the most prominent example of the equipment used for contact inspection of parts. When used with CIM these machines are controlled by CNC. A typical three-dimensional measuring machine consists of a table, which holds the parts in a fixed position and movable head, which holds sending probe. The probe can be moved in directions corresponding to the X, Y and Z coordinates. For manual operation, the control unit is provided with joysticks, or other devices which drive X, Y and Z servomotors (AC/DC).

During operation, the probe is brought into contact with the part surface to be measured and the three coordinate positions to a high level of accuracy. Typical accuracies of these machines are in the neighborhood of ± 0.004mm with a resolution of 0.001m. The measuring accuracy of a typical CMM is quoted $2.6 \pm L/100$micrometers,where L is the measured length in mm.

OPERATING SYSTEM

An operating system is a software in CNC that is used to control both the hardware and software of the system. There are different types of operating systems are used and they are decided by the type of manufacturer of the CNC machine.

- ECS
- LECS
- NUM
- SELCA
- MARPOSS
- Z-16
- FANUC
- FAGOR
- FIDIA
- DECKEL
- SINUMERIC
- HINUMERIC
- HEIDENHAIN
- GILDEMEISTER

Among these FANUC is the most widely used type of control system.

G & M CODES

There are two types of codes used for CNC programming. They are called G-Codes and M-Codes. G-code deals with the geometry of the hardware, for example, straight cutting developments, penetrating tasks and determining the units of estimation. While M-codes deals with the configuration of the machine tools Such as On/off commands and bringing back the machine to the origin or the cutting point.

LIST OF G CODES

G00 – Linear rapid position

G01 – Linear interpolation.

G02 – Circular interpolation clockwise.

G03 - Circular interpolation anti-clockwise.

G04 – Dwell

G05 – High precision counter control.

G07 – Imaginary axis designation.

G09 – Exact stop check.

G10 – Program data input.

G17 – XY plane selection.

G18 – YZ plane selection.

G19 – ZX plane selection.

G20 – Program in inches.

G21 – Program in mm.

G28 – Return to home position.

G30 – Return to secondary home position.

G33 – Constant pitch threading.

G40 – Tool radius compensation off.

G41 – Tool radius compensation left.

G42 – Tool radius compensation right.

G43 – Tool height offset compensation negative.

G44 – Tool height offset compensation positive.

G49 – Tool length offset compensation cancel.

G50 – Define maximum spindle speed.

G52 – Local coordinate system.

G54 – Work coordinate system.

G61 – Exact stop check.

G64 – Default cutting mode.

G69 – Turn off coordinate system.

G70 – Fixed cycle repeating for roughing.

G71 – Fixed cycle repeating for finishing.

G73 – Peck drilling.

G74 - Face grooving or chip break drilling; Lathe

G75 - OD groove pecking; Lathe

G76 - Fine boring cycle; Mill

G80 - Cancel cycles; Mill and Lathe

G81 - Drill cycle; Mill and Lathe

G82 - Drill cycle with dwell; Mill

G83 - Peck drilling cycle; Mill

G84 - Tapping cycle; Mill and Lathe

G85 - Bore in, bore out; Mill and Lathe

G86 - Bore in, rapid out; Mill and Lathe

G87 - Back boring cycle; Mill

G90 - Absolute programming

G91 - Incremental programming

G92 - Reposition origin point; Mill

G92 - Thread cutting cycle; Lathe

G94 – feed rate; Mill

G95 - Per revolution feed; Mill

G96 - Constant surface speed control; Lathe

G97 - Constant surface speed cancel

G98 - Per minute feed; Lathe

G99 - Per revolution feed; Lathe

LIST OF M CODES

M00 - Program stop; Mill and Lathe

M01 - Optional program stop; Lathe and Mill

M02 - Program end; Lathe and Mill

M03 - Spindle on clockwise; Lathe and Mill

M04 - Spindle on counterclockwise; Lathe and Mill

M05 - Spindle off; Lathe and Mill

M06 - Tool change; Mill

M08 - Coolant on; Lathe and Mill

M09 - Coolant off; Lathe and Mill

M10 - Chuck or rotary table clamp; Lathe and Mill

M11 - Chuck or rotary table clamp off; Lathe and Mill

M19 - Orient spindle; Lathe and Mill

M30 - Program end, return to start; Lathe and Mill

M97 - Local subroutine call; Lathe and Mill

M98 - Subprogram call; Lathe and Mill

M99 - End of sub program; Lathe and Mill

LETTER ADDRESS COMMANDS (TAB 1)

Letter Address	**Description**	**Refers to**
A	Angular Value about the X-axis. Measured in degrees	Axis nomenclature
B	Angular Value about the Y-axis. Measured in degrees	Axis nomenclature
C	Angular Value about the Z-axis. Measured in degrees	Axis nomenclature
D	the tool diameter or radius used for cutter compensation	Cutter compensation Parameter
E	second feed function	accuracy required when cutting a corner
F	Feed word (code)	Feed words
G	Preparatory word (code)	G-code Words
H	Unassigned/specifying for tool height compensation	
J	Interpolation parameter or thread lead parallel to the Y-axis	Circular interpolation and threading
K	Interpolation parameter or thread lead parallel to the Z-axis	Circular interpolation and threading
M	Miscellaneous or auxiliary function	Machine Control Codes
N	Sequence number	Program Line numbers
O	Sequence number for secondary head only	Indicates Program Number
P	P address character is used for a dwell time	Delay of time

Q	character is used in canned cycles	Depth specification
R	used in canned cycles or circular interpolation	Axis nomenclature
S	Spindle-speed function	Spindle speed
T	Tool Change function	Tool function
U	Secondary-motion dimension parallel to X	Axis nomenclature
V	Secondary-motion dimension parallel to Y	Axis nomenclature
W	Secondary-motion dimension parallel to Z	Axis nomenclature
X	Dimension of Tool movement in X direction	Axis nomenclature
Y	Dimension of Tool movement in Y direction	Axis nomenclature
Z	Dimension of Tool movement in Z direction	Axis nomenclature

MANUFACTURING PROCEDURE

The control must be provided with all the necessary information and instructions before work can begin. The following procedure for programming can be followed:

1. Select reference point, program zero
2. Determine coordinates (absolute or incremental (chain) dimensions)
3. Prepare working plan step by step
   a. Tool motions
   b. Feed rates
   c. Spindle speeds
   d. Tools used
   e. Coolant supply
4. Write program - Translating operation steps into programming language.
5. Key in program
6. Test and Edit program
7. Start Auto Cycle

MACHINES

DIFFERENT MACHINES

The following are the machines generally used in industry (TAB 2)

S. No.	Machine	Model	Controller	Traverse (mm)			Accuracy +/- (mm)	Repeatability +/- (mm)
				X	Y	Z		
1	Vertical machining center	VR- 3A Mitsui Seiki 1984	FANUC-6M	700	450	450	0.010	0.005
2	Vertical machining center	MCV-820	FANUC-6M	1600	820	700	0.010	0.005
3	LASER	Marko-0-2017	MARKO	500	500	400	0.060	0.060
4	Engraving machine	PASO 2011	ECKELMANN	600	500	300	0.01	0.05
5	CNC Lathe	AL-2A Mori Seiki 1987	FANUC-10T	480	-	310	0.015	0.005
6	CNC Lathe	SL-2A Mori Seiki 1987	FANUC-11T	57	-	510	0.015	0.005
7	Wire cut EDM	Electronica Ultra cut	5 Axes "ELPLUS"	1000	750	300	0.010	0.005

S.No.	Machine	Model	Control	Traverse (mm)			Rotary Table Diameter (mm)	Positional Accuracy +/- (mm)
				X	Y	Z		
9	Universal Machining Center	Deckel Maho "DMU80P"	5 Axes "Heidenhain ITNC 530"	800	800	800	900	0.007
10	Universal Machining Center	Deckel Maho "DMU 60 MonoBlock"	5 Axes "Heidenhain ITNC 530"	630	560	560	600	0.010
11	Universal Milling and turning machine	Deckel Maho "DMC 160 FD"	5 Axes "Heidenhain Mill plus it"	1600	1250	1000	1600	0.011
12	Tool and cutter grinder	Helitronic power	5 Axes "Walter"	460	320	660	D = 320 L = 350	0.010

13	Horizontal machining center	Deckel Maho “DMC 60H”	4 Axes “Siemens 840D”	600	500	560	400 x 400	0.008
14	Turn mill center	Deckel Maho “CTX 40052”	3 Axes “Fanuc 21 ITB”	D = 260 L = 600	12 stations (6 driven)	0.010		

TYPES OF MACHINES

CNC machining is a manufacturing process in which pre-programmed computer software dictates the movement of factory tools and machines. This process can be used to control a range of complex machinery from grinders and lathes to mills and routers. Three dimensional cutting tasks can be accomplished in a single set prompt.

Open/Closed loop machining systems:

Position control is determined by these loops. Signaling runs in a single direction between controller and motor in open loop whereas in closed loop controller can receive feedback which makes error correction possible. Thus closed loop can rectify can rectify irregularities.

The earliest numerical control machines date to the 1940s when motors were first employed to control the movement of pre-existing tools. As technologies advanced, the mechanisms were enhanced with analog computers, and ultimately with digital computers, which led to the rise of CNC machining.

The vast majority of today’s CNC arsenals are completely electronic. Some of the more common CNC-operated processes include ultrasonic welding, hole-punching and laser cutting. The most frequently used machines in CNC systems include the following:

CNC MILLS

CNC mills are capable of running on programs comprised of number- and letter-based prompts, which guide pieces across various distances. The programming employed for a mill machine could be based on either G-code or some unique language developed by a manufacturing team. Basic mills consist of a three-axis system (X, Y and Z), though most newer mills can accommodate three additional axes.

Conventional type:

- In conventional milling the cutter rotates against the direction of the feed while during climb milling, the cutter rotates with feed. The conventional type is the ancient and widely used one.
- In conventional type the material is removed by direct contact between the tool and job.

Non – Conventional type:

- In non- conventional type the material is removed by applying some energy to job. Material is removed by static electricity or magnetic energy etc.
- It is the advanced type of cutting and used only in case of necessary.

LATHE

In lathe machines, pieces are cut in a circular direction with indexable tools. With CNC technology, the cuts employed by lathes are carried out with precision and high velocity. CNC lathes are used to produce complex designs that wouldn't be possible on manually run versions of the machine. Overall, the control functions of CNC-run mills and lathes are similar. As with the former, lathes can be directed by G-code or unique proprietary code. However, most CNC lathes consist of two axes — X and Z.

PLASMA CUTTERS

In a plasma cutter, material is cut with a plasma torch. The process is foremost applied to metal materials but can also be employed on other surfaces. In order to produce the speed and heat necessary to cut metal, plasma is generated through a combination of compressed-air gas and electrical arcs.

ELECTRIC DISCHARGE MACHINES

Electric-discharge machining (EDM) — alternately referred to as die sinking and spark machining — is a process that molds work pieces into particular shapes with electrical sparks. With EDM, current discharges occur between two electrodes, and this removes sections of a given work piece.

WATER JET CUTTERS

In CNC machining, water jets are tools that cut hard materials, such as granite and metal, with high-pressure applications of water. In some cases, the water is mixed with sand or some other strong substance. Factory machine parts are often shaped through this process.

MACHINE OPERATION

For manufacturing the job in this project, VR - 3A Mitsui Seiki 1984 Vertical machining centre with FANUC 6M control system was used. The following are the controls present on the machine and their functions.

OFFSET:

It is the distance between machine origin and work origin. It is to be calculated correctly and noted in the machine to avoid unnecessary cutting.

OPTIONAL LOCK:

It should be in tape mode for actuating spindle and other controls as per the program. It won't work even though it is given in the program but is in OFF condition.

DRY RUN

Move the tool fast while the time of execution. For example, if the cutting is completed in particular depth it will return again while it is returning it moves fast.

PRP

It is used to start the control of the machine after it is started without executing PRP we can't actuate the machine controls.

POS

POS provides the position of the tool its relative, actual and absolute position.

PROG:

This shows the position of the material from the machine to the reference point. It calculates all the programs including the subprograms the main program.

EOB:

It is called End of the Block which is necessary to separate one program line to another of not this is used the lines will be executed as a whole.

INSERT:

It is used to insert the values in between the lines if necessary in the machine.

ATC:

We can operate the machine manually most importantly in tool changing case, this will find importance when machine stop working accidentally.

RESET:

It helps the machine to go to the initial position. It is one of the manual operation done when it is necessary.

ALTER:

We can change any value in the program while it is executing and run it again. Executing line in the program is shown in the screen.

AUTO:

Pressing this button enables the CNC programs stored in the memory to be executed for automatic operation. When the Cycle Start button is pressed and this mode is active, automatic operation will occur.

SINGLE BLOCK:

The execution of a SINGLE BLOCK (SINGLE BLOCK) of information is initiated by pressing this button to turn it ON. Each time the CYCLE START button is pressed, only one block of information will be executed. This switch can also be used if you intend to check the initial performance of a new program on the machine or when the momentary interruption of a machine's work is necessary.

RESTART:

The 'RESTART' button allows you to restart automatic operation at a specific place in the program by entering a desired line sequence number. This sometimes becomes necessary after tool breakage or some type of collision.

PRG STOP: -

When a Program Stop is commanded in the program by the program word M00, automatic operation is stopped and the LED on this button is turned on. This button does not have an ON/OFF function that affects the program stop.

SPDL CW: -

By pressing the SPDL CW button while in one of the operation modes REF, JOG, INC, or HANDLE, the spindle will start rotation in the clockwise (CW) direction. The spindle rev/min is adjusted by using the Spindle Override dial. When set at 100%, the spindle will rotate the last r/min commanded in the program.

Z-NEGLECT

Tool moves only in the X and Y axis while the Z axis is kept constant.

OPTIONAL BLOCK SKIP

It is used to skipping some block of the program and execute the rest of the program.

EDIT

Used to make changes in code

MDI

It takes us directly to the line of code which needs to be edited. the whole code need not be written again. for example, tool change.

CLNT CONTROLS

They are used to control coolant.

REMOTE

While using Remote, AUTO EDIT and MDI can't be used.

+, - CONTROLS

It changes the amount by which spindle moves by 1 rotation of the control.

SYS

Shows the logic of the system in binary.

GRAPH

Shows the graph of job movement

(Fig. of control (representation purpose) FIG. 2)

CNC SETTING

1) OFFSET - To make a new point in software or shifting any point in software from its actual position known as offset.

2) GEOMETRY OFFSET - Geometry offset is used for making job origin or job centre at a distance M/c home position. geometry offset is used for tool held in the turret each tool requires separate geometry offset.

3) WEAR OFFSET - Wear offset is used for shifting the tool from its actual position by giving the value in software, in CNC turning wear offset is used in X & Z axes.

4) WORKSHIFT OFFSET - Work shift offset is used for shifting the job origin from its actual position by giving the value in software.

TOOLS

TOOLING MATERIALS

Carbide tools:

These are one of the inexpensive tools used for cutting, milling for low speed milling operations. Cemented carbide (tungsten carbide) is prepared using powder metallurgy. it is sintered at 2500 deg celsius. carbide particles are bound to cobalt and it becomes as hard as a diamond. hence it is very brittle too. they are available at a lesser grade if they are more tough. These metals lose their hardness at high temperatures so they cannot be used at high temperatures. They possess good machinability.

their wear resistance can be improved by coating TiC, TiN and AlO by plasma vapour method.

Some of the recent coated cutting tools include polycrystalline diamond (PCD). it has hardness of a diamond and friction coefficient of teflon. hence it has longer tool life.

Carbide tools are used for performing operations like twist drilling, milling, turning, cutting for the materials like brass, aluminium, steel, glass etc.

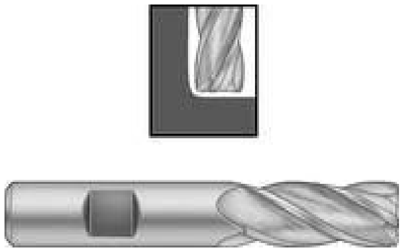
<u>HSS: -</u>

These also have good machinability but are less efficient and accurate compared to Carbide tools. These are less expensive and made of ductile materials. They are used for performing operations like drilling, chamfering etc.\

The following are the different types of tools.

Rounded Edge End Mills or Flat end mills

Rounded edges on the tip of the flutes reduce chipping and increase the life of the tool. Also known as corner radius end mills and bull nose end mills. they work well for making molds. Use on cast iron, cast steel, and heat-treated steel. FIG 3

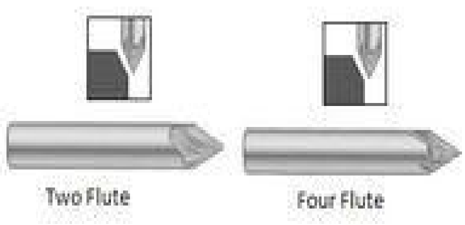


End Mills

End mills are tools which have cutting teeth at one end, as well as on the sides, they are used for a variety of operations including facing an edge, and cutting slots or channels. FIG 4

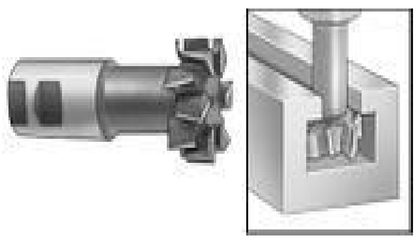
Chamfer End Mill:

Used to make beveled and angled cuts without the need for hand deburring. Two flutes provide better chip clearance, while four flutes produce a finer finish. FIG 5

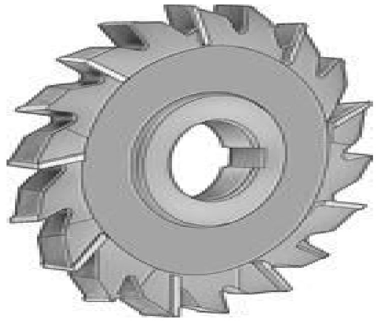


T-Slot Cutters:

Used for milling T-slots in machine tool tables, indexing tables, and other work holding surfaces and products. Before using the cutter, a slot in the work surface needs to be milled so that the neck of the cutter can enter the cut. FIG 6

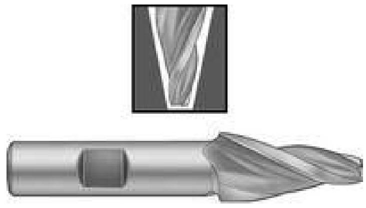


Staggered-Tooth Milling Cutters:



With peripheral and side teeth, cutters remove chips for consistent cutting width. Cutters are for deep slots and other applications requiring maximum chip clearance. Also known as side-milling cutters, they have alternating right- and left-hand helical teeth to remove more metal at higher feeds/speeds than straight-tooth cutters. If you interlock two or more cutters of the same diameter so you can customize the thickness of the cutter to match the slot width you require.

Tapered End Mills:



Used to machine angled slots in dies and molds. Also known as taper degree end mills.

Fly Cutter;

Works well to remove material in one pass on a larger surface, this tool can be hard on the machine when working with hard metals because of the distance the cutter is away from the center of the tool.

Drill:

Drill bits are cutting tools used to remove material to create holes, almost always of circular cross-section. Drill bits come in many sizes and shapes and can create different kinds of holes in many different materials. In order to create holes' drill bits are usually attached to a drill, which powers them to cut through the workpiece, typically by rotation.

Central drill:

It is a primary drill made before the normal drill to indicate the coordinate exact where the drill to be made. The material used for these type of drill is different from normal drill materials.

OPERATIONS

Milling:

Milling is the process of cutting away material by feeding a workpiece past the tool. The tool remains stationary only work piece moves in the operation. An end mill is the term used for most common types of milling cutters.

Drilling:

In drilling operations, the estimation of metal cutting is about 75 % of material is removed. Right circular cylindrical holes are created by using a twist drill. Coolant application is difficult in drilling as cutting front is embedded in the work piece. So the coolant spray mist is applied or coolant is applied through drill bit shaft. The chips must exit through the flutter to the outside. chip exit can be a problem when chips are larger or continuous .BW drill has a greater strength when compared to conventional drill and thus can be used for bulk material removal.

Tapping:

Tapping is used to produce the internal threads in drilled holes. A tap is multi fluted cutting tool with cutting edges on each blade in the shape of threads. It is a form of cutter reproducing the shape of its cutting edge in the work. Taps are made of either carbon steel or high speed steel. Most common types of hand taps are Taper taps, Plug taps, Bottoming taps.

Broaching:

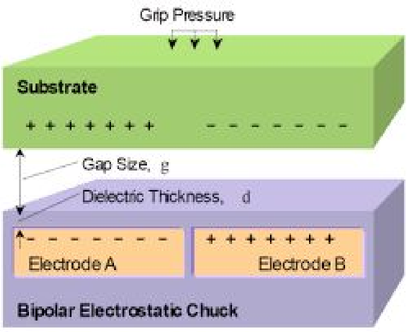
A broach is a series of progressively taller chisel points mounted on a single piece of steel. This is used to enlarge a circular hole into a large non circular shape such as square or other desired shape. Another use of broach is to cut splines or a square keyway on objects such as gears, shafts, pulleys etc. Amount of material removed by broach tool varies with material being cut.

CLAMPING

Vacuum clamping:

- Used for thin plates to avoid damage
- Light clamping.
- Clamping face is circumscribed by rubber.

Electrostatic Clamping: -



- Workpiece is charged with static electricity with polarity opposite to polarity on the chuck face.
- Opposite attracting polarities of work piece and chuck face develop the clamping force.

- Workpiece must be cleaned and dry force good electrostatic clamping.

Fluid power clamping:

- Fluid power clamps are generally actuated by cylinders.
- Fluid power clamping is a clamping fixture with clamping nut attached to cylinder ram.
- Feeding pressurized fluid through port pulls ram downwards and presses clamp against work piece,
- There will be an unpressurised discharge line for unclamping.

Pneumatic clamping:

- Compressed air is used as fluid for power transmission and application.
- Large centralized compressor supplies pressurized air.
- Where speed variations are unacceptable.
- If load resistance increases speed of pneumatic device decreases.

Hydraulic clamping:

- It has more advantages than pneumatic clamping.
- Hydraulic oils are incompressible.
- Variation in load does not cause much speed variation.

Cam clamping:

- These clamps have an eccentric cam that locks against the side of the workpiece by fastening a screw.
- Clamping levers are intended for manual adjustment by fixation tasks and clamping applications. Cam levers are ideal for quick fixation. The product group is characterized by a wide variety of designs, materials and colors. These classic elements find use in machine construction, tool making and plant construction.

Pull clamps:

- The pull clamps have a clamping pin or screw that is inserted through a hole in a workpiece into a hole on the clamp. When the clamp is engaged the clamping pi is pulled down and work piece is clamped into place.

Push clamps:

- These type of clamps are machining clamps that have a piston on the jaw that pushes and clamps against the workpiece when clamp is actuated.

Retracting clamps:

- These type of clamps are a machining clamps that have a retractable clamping arm for fast part loading and unloading.

Swing clamps:

- They are fixturing and set up clamps, that are used to secure a jig or fixture. They swing out and away from clamped part to achieve clearance for part removal.

Toe clamps:

- They are low profile clamps that butt upon the side of workpiece. As they are tightened a wedge moves the jaw downward and forward to provide a positive force against the side of the workpiece.

Fixtures:

- Fixtures are made for inspection and assembly works. Moreover, fixtures are used for castings and forgings which are rough and irregular in shape. With the use of locators and proper clamps, handling of those jobs will be made easy in fixtures than any other standard work holding devices.
- Some of the type of the fixtures used are assembly fixtures, form fixtures, milling fixtures, reciprocating fixtures etc.

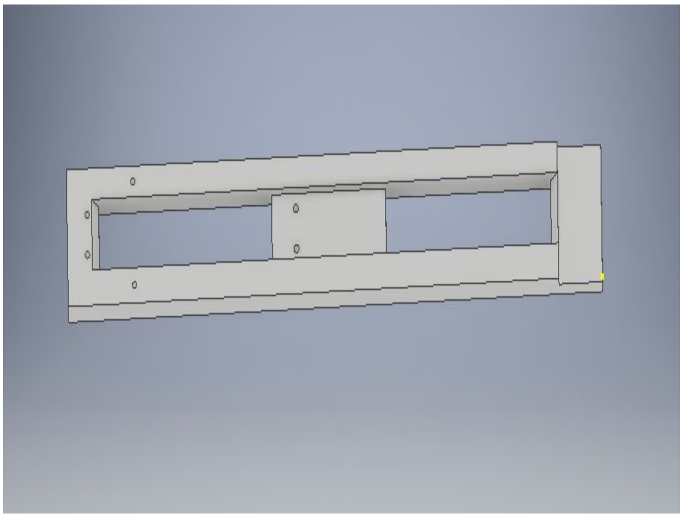
Jigs:

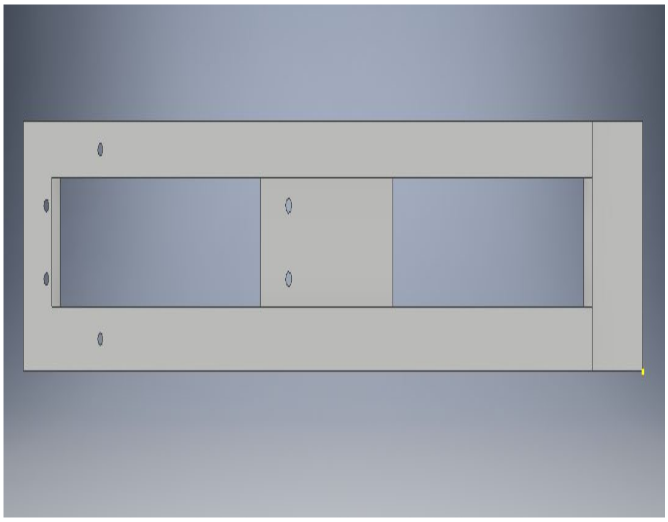
- They can be used for both fixture and tool path guidance for the tool. For example, if we want to cut the job in an angle then a specific jig is placed to the above of the job which will guide the tool though it is not used for cutting it only specify the movement of the tool.

CHAPTER 3

MANUFACTURING THE JOB

The top and isometric views of the job (not to scale)





(Due to copyright issues, many operations done on the actual job have not been reproduced here on the picture and the code. the following code is only for some of the operations done on the top face of the job)

PROGRAM FOR THE JOB

```
%
02345
( Program name - bottom)
G0 G15 G17 G40 G49 G69 G80 G90
G54 G30 G91 Y0 Z0
T2 M6 ( 10. FLAT END MILL)
G90 G0 X0 Y0
G43 H2 Z100. M1
S1200 M3 Z50.
M8 D8
```

(T SHAPE POCKET DEPTH 0 - 5)

```
G0 G90 X277.2 Y22.9
Z2.
#102 = -0.
#103 = 100
#104 = 1200
#105 = -0.5
M98 P3450 L10
G0 Z2.
```

(LEFT POCKET 101* 29.5 DEPTH 5 - 8)

```
G0 G90 X277.2 Y22.9
Z2.
#102 = -5.
#103 = 100
```

#104 = 1200
#105 = -0.5
M98 P3451 L6
G0 Z2.

(RIGHT POCKET 101* 29.5 DEPTH 5 - 8)

G0 G90 X182.05 Y29.65
Z2.
#102 = -5.
#103 = 100
#104 = 1200
#105 = -0.5
M98 P3452 L6
G0 Z2.

(TOOL CHANGE TO CENTRE DRILL)

Z20. M5.
Z100. M1 M9
G91 G30 Y0 Z0
T11 M6
G0 G90 X0 Y0
G43 Z100. H11 M1
Z20. S2500 M3
M8

(CENTRE DRILL)

G90 G98 G81 Z-0.5 R2. F100.
M98 P3456
G80 G0 Z5.

(TOOL CHANGE TO 2”)

Z20. M5.
Z100. M1 M9
G91 G30 Y0 Z0
T10 M6
G0 G90 X0 Y0
G43 Z100. H10 M1
Z20. S2500 M3
M8

(2 HOLES OF 2.5 M DIA 8 DEEP)

G0 G90 X40.4 Y38.5
G98 G83 Z-8. R10. F100.
Y3.
G90 X40.4 Y38.5
G80 G0 Z5.

(TOOL CHANGE TO 2.5”)

Z20. M5.
Z100. M1 M9
G91 G30 Y0 Z0
T9 M6

```
G0 G90 X0 Y0
G43 Z100. H9 M1
Z20. S2500 M3
M8
```

(2 HOLES 3 M DIA 8 DEEP)

```
G0 G90 X17.1 Y10
G99 G83 Z-8. Q.5 R10. F100. K0
Y26.
G90 X17.1 Y10.
G80 G0 Z5.
```

(**TOOL CHANGE TO 2.8")**

```
Z20. M5.
Z100. M1 M9
G91 G30 Y0 Z0
T8 M6
G0 G90 X0 Y0
G43 Z100. H8 M1
Z20. S2500 M3
M8
```

(2 COUNTER SINK HOLES 2.8 DIA 8 DEEP)

```
G0 G90 X137.9 Y28.3
G99 G83 Z-8. Q.5 R10. F100. K0
Y13.2
G90 X137.9 Y28.3.
G80 G0 Z5.
```

(**TOOL CHANGE TO 8. FLAT END MILL)**

```
Z20. M5.
Z100. M1 M9
G91 G30 Y0 Z0
T3 M6
G0 G90 X0 Y0
G43 Z100. H3 M1
Z20. S2500 M3
M8
```

(**T SHAPE CONTOUR FINISH DEPTH 0 - 5)**

```
G0 G90 X37.4 Y20.75
Z2.
#102 = -0.
#103 = 100
#104 = 1200
#105 = -0.5
M98 P3453 L10
G0 Z2.
```

(**LEFT 101*29.5 CONTOUR FINISH DEPTH 5 -8)**

```
G0 G90 X79.9 Y27.4
Z2.
```

#102 = -5.
#103 = 100
#104 = 1200
#105 = -0.5
M98 P3454 L6
G0 Z2.
(RIGHT 101*29.5 CONTOUR FINISH DEPTH 5 -8)

G0 G90 X184.2 Y20.75
Z2.
#102 = -5.
#103 = 100
#104 = 1200
#105 = -0.5
M98 P3455 L6
G0 Z2.

(TOOL CHANGE TO 10" CHAMFER TOOL)

Z20. M5.
Z100. M1 M9
G91 G30 Y0 Z0
T3 M6
G0 G90 X0 Y0
G43 Z100. H3 M1
Z20. S2500 M3
M8

(CHAMFER 10 MM DEPTH 5)

G0 G90 X34.4 Y25.5
Z10.
G1 Z-5. F200.
G41 D3 Y35.5 F1500
X24.4
Y6.
X34.4
G40 Y16.
G0 Z50.

(TOOL CHANGE TO 4" CHAMFER TOOL)

Z20. M5.
Z100. M1 M9
G91 G30 Y0 Z0
T4 M6
G0 G90 X0 Y0
G43 Z100. H4 M1
Z20. S2500 M3
M8

(CHAMFER 10 MM DEPTH 8)

G0 G90 X275.2 Y10
Z10.
G1 Z-8. F1000.
G41 D4 Y6. F2000
X279.2

Y35.5
X275.2
G40 Y31.5
G0 Z50.

(PROG END SEQUENCE)

M9
S500 M5
Z100. M1
G91 G30 Y0 Z0
M30

03450
G1 Z#102 F#103
G1 X42. F#104
Y18.6
X277.2
X295.9
Y22.9
X277.2
Y30.4
X34.5
Y11.1
X277.2
X302.3
Y.1
X303.4
Y41.4
X302.3
Y30.4
X277.2
G0 Z5
X277.2 Y22.9
#102 = #102 + #105
M99

03451
G1 Z#102 F#103
X43.25 F#104
Y19.85
X116.55
Y21.65
X124.8 Y29.9
X35
Y11.6
X124.8
Y29.9
G0 Z5
X116.55 Y21.65
#102 = #102 + #105
M99

03452
G1 Z#102 F#103
X271.35 F#104
Y11.85
X182.05

```
Y29.65
X190.3 Y21.4
X263.1
Y20.1
X190.3
Y21.4
G0 Z5
X271.6 Y29.9
Z10
G1 Z#102 F#103
X181.8 F#104
Y11.6
X271.6
Y29.6
G0 Z5
X182.05 Y29.65
#102 = #102 + #105
M99

03453
G1 Z#102 F#103
G41 X29.4 F#104
Y6.
X277.2
X297.2
Y-5
X308.5
Y46.5
X297.2
Y35.5
X277.2
X29.4
Y20.75
G40 X37.4
G0 Z2
X37.4 Y20.75
#102 = #102 + #105
M99

03454
G1 Z#102 F#103
G41 Y35.4 F#104
X29.5
Y6.1
X130.3
Y35.4
X79.9
G40. Y27.4
G0 Z2
X79.9 Y27.4
#102 = #102 + #105
M99

03454
G1 Z#102 F#103
G41 Y35.4 F#104
X29.5
Y6.1
```

```
X130.3
Y35.4
X79.9
G40 Y27.4
G0 Z2
X79.9 Y27.4
#102 = #102 + #105
M99

03455
G1 Z#102 F#103
G41 X176.2 F#104
X277.2
Y35.5
X176.2
Y20.75
G40 X184.2
G0 Z2
X184.2 20.75
#102 = #102 + #105
M99

03456
G90
X137.9 Y28.3
Y13.2
X17.1 Y10
Y26
X40.4 Y38.5
Y3
G80
M99
```

FINISHING OF THE JOB

Since the job is machined under cool working conditions (since coolant is used) , unwanted stresses get accumulated in the job due to which on application of load, the job may fail. hence the CNC machined job is sent to the heat treatment cell, where stress relieving is done.
The Heat treatment cell contains 3 types of furnaces -

1) Muffle furnace
2) Oven Furnace
3) Vacuum Furnace

The Furnace is selected based on various factors such as Max temperature the job can withstand, weight and dimensions of job, job surface. Vacuum furnace is the most advanced and computer operated furnace and hence there is no rust formation, corrosion, damage to the shape and changes in dimensions. The heating takes place in a controlled manner and cooling also happens within the furnace.

The Finishing of the jobs, here, is done by various chemical processes like phosphating, chemical blackening, thermal blackening, stainless steel passification, mild steel passification, copper oxidation. and electrochemical processes like chromatization, electroplating, anodizing, electropolishing, cataphorous varnishing, Hard anodizing.
The appropriate process is chosen according to the material to be deposited (like silver, gold, nickel, cadmium, tin bismuth alloy, tin zinc alloy, copper) cost factors, extent to which surface needs to be finished.

The job is finally sent to the painting cell and then moved to the assembly line, where it is assembled along with the other parts.

CHAPTER 4

RESULT AND CONCLUSION

Through this project work, working of the software MASTERCAM, CNC Programming, CNC Machining, machine operations and Science of tools and tool operations have been studied about in detail and the underlying concepts have been applied for the manufacturing of a mounting bracket of avionic components.

The Shop floor at HAL Hyderabad, manufacturing department which contains various cells such as the heat treatment cell, surface finish cell, metrology cell, methods and planning cell, QC cell, Fabrication hanger which were all involved with the manufacturing of the concerned job, have all been keenly observed and practical insight into theoretical concepts has been provided.

CONCLUSION

The importance of conventional lathes and milling machines can never be undermined. these machines have played an important role in bringing about the industrial revolution and have laid the foundations. but the bringing about of the new technology in the new era is also equally important. for this we need to increase the purchase and usage of CNC machines to improve the quantity and quality of production in the industry, and limit conventional machines to small quantities.

CHAPTER 5

REFERENCES

1) CNC Programming : Principles and Applications by Mike Mattson
2) CNC Programming a practical approach by Jon stenerson and Kelly Curran
3) **https://ielm.ust.hk/dfaculty/ajay/courses/ieem215/labs/9_masterCAMintro.pdf**
4) **https://www.mastercam.com/news/blog/**
5) **https://www.slideshare.net/PradeepKumar1691/cnc-training-report**
6) **https://hal-india.co.in/Products/M__54**
7) www.wikipidea.org